Computer Simulation of Tidal Currents in GHESHM Canal

SAEED-REZA SABBAGH-YAZDI, SAEED SHARBATI
KN.Toosi University of Technology,
Department of Civil Engineering,
1346 Valiasr St. Tehran, Iran

NIKOS E. MASTORAKIS
Military Insitutes of University Education (ASEI)
Hellenic Naval Academy
Terma Chatzikyriakou 18539, Piraues, GREECE

Abstract:
The flow pattern in the GHESHM canal is affected by two open boundary conditions at inlet and outlet of the canal in which are formed by tidal currents in the PERSIAN Gulf . In this paper, Hydrodynamic simulation of tidal currents in GHESHM canal due to tidal fluctuations in HURMOZ Strait is presented. The mathematical model utilized consists of depth averaged equations of continuity and motion in two-dimensional horizontal plane which considering hydrostatic pressure distribution. The finite volume method is applied for converting the governing equations into discrete form for unstructured overlapping cell vertex control volumes. In order to reduce the unwanted errors during model running, the artificial viscosity was inserted to the source code of the model. Finally, the performance of the computer model to simulate tidal flow in a geometrically complex domain is examined by simulation of tidal currents.

Key-Words: Tidal Currents, GHESHM Canal, Shallow Water Equations, Unstructured Finite Volumes, NASIR Software.

1 Introduction

The computer simulation of complicated marine environment problems have become one of the interesting areas of the research works by development of efficient and accurate numerical methods suitable for the complex flow domain. The control over properties and behaviour of fluid flow and relative parameters are the advantages offered by computational fluid dynamics (CFD) which make it suitable for the simulation of the applied engineering problems [1].

Water free surface treatment is one of the major difficulties in numerical solution of three dimensional water flow patterns. Some of the models apply a normalized coordinate (sigma coordinate) system in vertical direction to solve three dimensional flow equations. Shallow water flow solvers are models with low computational costs and suitable for the cases in which vertical component of the flow velocities are negligible [2].

In this paper the a version NASIR (Numerical Analyzer for Scientific and Industrial Requirements) software which is developed for finite volume solution of shallow water equations is assessed for simulation of tidal flow patterns in a canal in variable bathymetry. Therefore, special treatments are considered for moving coastal boundaries due to the flood and drying of the tidal flats and mangrove forests.

2 Hydrodynamic equations

The depth-integrated continuity and momentum equations of free surface water flow are:

$$\frac{\partial h}{\partial t}+\frac{\partial (hu)}{\partial x}+\frac{\partial (hv)}{\partial y}=0 \qquad (1)$$

$$\frac{\partial (hu)}{\partial t}+\frac{\partial (hu^2)}{\partial x}+\frac{\partial (huv)}{\partial y}= -gh\frac{\partial \eta}{\partial x}-\frac{\tau_{bx}}{\rho_w}+hv_{Th}(\frac{\partial u}{\partial x}+\frac{\partial u}{\partial y}) \qquad (2)$$

$$\frac{\partial (hv)}{\partial t}+\frac{\partial (huv)}{\partial x}+\frac{\partial (hv^2)}{\partial y}= -gh\frac{\partial \eta}{\partial y}-\frac{\tau_{by}}{\rho_w}+hv_{Th}(\frac{\partial v}{\partial x}+\frac{\partial v}{\partial y}) \qquad (3)$$

In the above equations, the abbreviations are known as, t-time, x and y-horizontal Cartesian coordinates; h-flow depth; u and v-depth-averaged flow velocities in x and y directions; zs-water surface elevation ($z_s = h + z_b$); and g-gravitational acceleration.

Bed friction global stresses in x and y $\tau_{bx}=\rho_w C_f u|U|$ and $\tau_{by}=\rho_w C_f v|U|$ directions are calculated using $C_f = gn^2/h^{1/3}$ (n represents manning coefficient). In the present work, the widely used depth-averaged parabolic turbulent model is applied, in which the eddy viscosity parameter is computed algebraic formulation $v_t = \theta\, hU_*$. In this formulation the bed friction velocity is defined as $U_* = [C_f(u^2+v^2)]^{0.5}$ and the empirical coefficient θ is advised between 0.3 and 1.0. This turbulence model is known suitable for depth averaged equations and has been used in some similar applications [3,4, 5].

3 Numerical Formulations

Considering the governing equations in form of an advection-diffusion type equation, W represents the variables using h flow depth, hu and hv the horizontal components of velocity. While, G^d and F^d are vectors of diffusive fluxes of W in x and y directions, respectively. The vector S contains the sources and sinks of the governing equations covering all algebraic terms.

$$\frac{\partial W}{\partial t}+(\frac{\partial F^c}{\partial x}+\frac{\partial G^c}{\partial y})=(\frac{\partial F^d}{\partial x}+\frac{\partial G^d}{\partial y})+S \qquad (4)$$

The governing equations are discretized by the application of cell vertex (overlapping) scheme of the finite volume method [6]. Considering proper techniques to convert diffusive fluxes into discrete form and source term treatment, this method ends up with the following formulation.

$$W_i^{t+\Delta t}=W_i^t-\frac{\Delta t}{\Omega_i}.\sum_{k=1}^{N_{sides}}[(\overline{F}^c\Delta y-\overline{G}^c\Delta x)-(F^d\Delta y-G^d\Delta x)]_k^t +\frac{\Delta t}{3}S_i^t \qquad (5)$$

Where Wi represents conserved variables at the centre of control volume □i. $\overline{F}^c$ and $\overline{G}^c$ are the mean values of convective fluxes on the control volume boundary sides. S_i^t is the source term which covers volumetric evaporations and body forces. The diffusive fluxes F^d and G^d are computed using a discrete formula of contour integral around the centre of the control volume boundary sides (using an auxiliary control volume).

The residual term, $R(W_i)=\sum_{k=1}^{N_{sides}}[(\overline{F}^c\Delta y-\overline{G}^c\Delta x)-(F^d\Delta y-G^d\Delta x)]_k^t$, consists of convective and diffusive part. In smooth parts of the flow domain, where there is no strong gradient of flow parameters, the convective part of the residual term is dominated. Since, in the explicit computation of convective dominated flow there is no mechanism to damp out the numerical oscillations, it is necessary to apply numerical techniques to overcome instabilities with minimum accuracy degradation. In the present work, the artificial dissipation terms suitable for the unstructured meshes are used to stabilize the numerical solution procedure. In order to damp unwanted numerical oscillations, a fourth order artificial dissipation term, $D(W_i)=\varepsilon\sum_{j=1}^{N_{edges}}\lambda_{ij}(\nabla^2 W_j-\nabla^2 W_i)$ is added to above algebraic formula in which λ_{ij} is a scaling factor and is computed using the maximum value of the spectral radii of every edge connected to node i ($1/256 \le \varepsilon \le 3/256$). Here, the Laplacian operator at every node i, $\nabla^2 W_i = \sum_{j=1}^{N_{edges}} (W_j - W_i)$, is computed using the variables W at two end nodes of edges (meeting node i). The revised formula, which preserves the accuracy of the numerical solution, is written in the following to following relation.

$$W_i^{t+\Delta t} = W_i^t - \frac{\Delta t}{\Omega_i}.\{R(W_i^t) - D(W_i^t)\} + \frac{\Delta t}{3} S_i^t \quad (6)$$

Δt is the minimum time step of the domain proportional to the minimum mesh spacing and maximum wave speed of the convective homogonous equations. [7]

4 Boundary Conditions
Two types of boundary conditions boundary conditions are applied in this work;

4.1 Flow Boundary Conditions
The tidal flow boundary condition (like outer boundary of the test case or HURMOZ strait of Persian Gulf application case) can be applied by imposing the water surface level fluctuations at tidal flow boundary. Surface water oscillating boundary condition was imposed at the inlet boundary as SHAHID-RAJAEE port (east boundary) and BASAEED port (west boundary). Regarding to the shortage of data for water level oscillation in west boundary through numerical simulation period, the simulation and prediction of water level of tides was applied by 36 tidal constituents of Iranian national cartography centre tide's poll. For achieving this data in the form water level fluctuations, harmonica analysis of MIKE21 software was used. For the east boundary, the tide gauge data of national cartography centre was applied.

4.2 Wall boundary Conditions
Free-slip velocity condition walls can be imposed where no flow passes through a vertical plane of the flow domain. This is the condition for straight borders of the first test case. These boundaries can be used to reduce the computational domain due to symmetric conditions in radial walls of the first test case. At these boundaries the component of the velocities normal are set to zero. Therefore tangential computed velocities are kept using free slip condition at wall boundaries.
The movement of the wall boundaries due to wet and drying of the tidal flats are simulated by imposing zero velocity at nodes in which the computed depth is less than a prescribed value. However, small water depths in coastal zones of the canal may give rise to the global bed shear stresses and reduce the computed velocities in these regions [8].

5 GEOMETRY modelling
The coastal boundary and the bed surface topography of the GHESHM is very irregular. A numerical model is not able to simulate the real world flow pattern unless the geometrical characteristics of flow domain are modeled precisely. Thus, the numerical flow solver should handle the geometrical complexities of the bed and boundaries of the flow domain. In order to overcome the problem, in the present work, attempt has made to solve the depth averaged hydrodynamic equations on unstructured finite volumes. Application of unstructured mesh facilitates considering the effects of geometrical irregularities of coasts [9].

The three dimensional surface of flow bed is modelled in two stages. In the first stage, horizontal geometry of the problem is modelled by definition of some boundary curves, (281 curves of coastal boundaries) and then, the domain is discretized with triangular unstructured mesh which contains 3599 nodes, and 10218 cells. In the second stage, for converting the two dimensional mesh into a three dimensional surface, the bed elevation of the flow domain is digitized at a number of points along some contour lines. Then, the bed elevation is set for the every node of the mesh by interpolation of the elevations of surrounding digitized points. Figure (1) shows the GHESHM Island location and figure (2) shows geometry of the GHESHM canal and its constructed triangular meshes within.

Figure1. Position of the GHESHM Canal between the GHESHM Island in the Iranian coast,

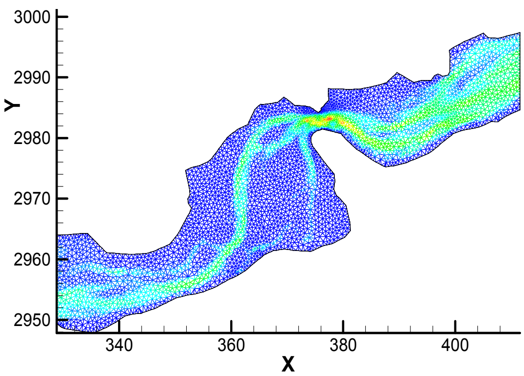

Figure2. Discrete geometrical model of GHESHM canal considering bottom elevations

6 Model results

As it noted before, the hydrodynamic model calculates the depth-averaged equations in the solving domain. For applying this, the observed data of water level oscillation at the east boundary (SHAHID-RAJAEE Port) and predicted water level oscillation at the west boundary (BASAEED Port) were considered as input data to the numerical domain.

The comparison on water level oscillation results between observed data at KAVEH Pier and numerical model was performed in a period of 48 hours (16-feb-2002 and 17-feb-2002) is shown in figure (3). It worth noting that, the velocities comparison between model results and domain gathered data, alludes to the model competency in tidal currents simulation in the present area. Also, Velocity evaluation on the model outcomes and observed data is shown in figure (4).

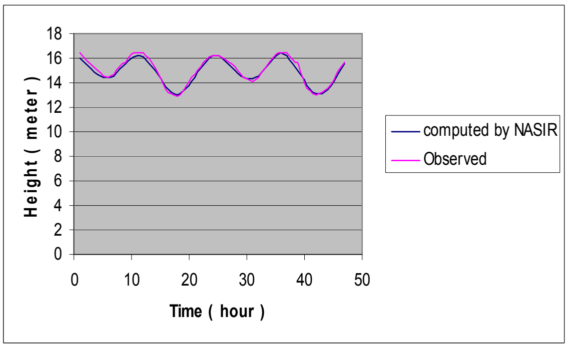

Figure 3. Curves of water level oscillation obtained from the numerical model and observed data at KAVEH Pier in a 48 hours period.

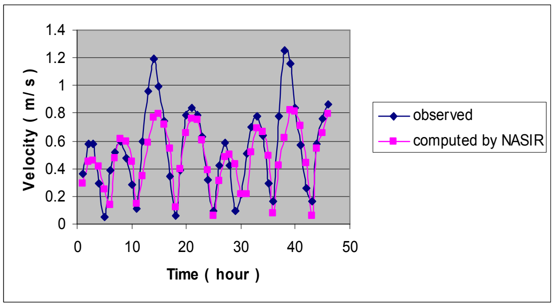

Figure 4. Curves of flow velocity obtained from the numerical model and observed data at KAVEH Pier in a 48 hours period.

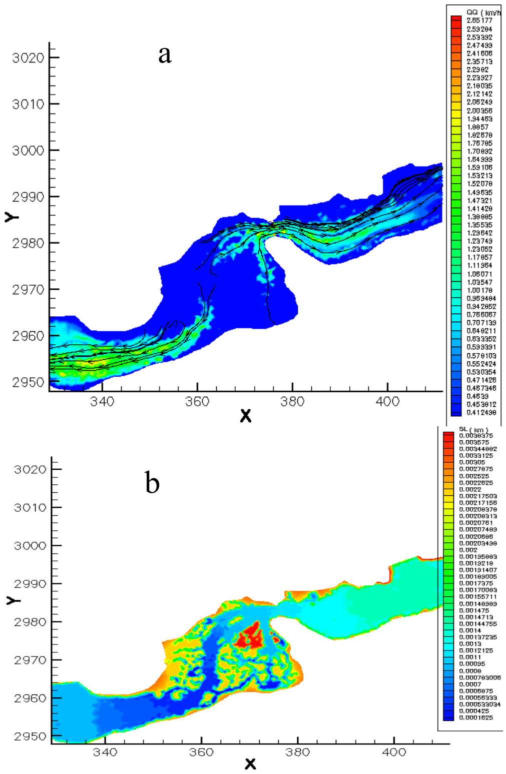

Figure 5. Color Coded map of the GHESHM canal; a) velocity b) water elevation alteration

7 Results Discussion

As it is apparent from the graphs in figures 3,4 and 5 which reveal the results for water elevation, velocity and flow direction, numerical model results show a fair correlation between observed data, but the amount of error is different in flow elevation, velocities and flow direction. For instance the outcomes of model and observed data related to the tidal water level, express an excellent correlation (an average error of 1.15 percent) in which comparison between velocities demonstrates an average error around 8.3 percent. So the model could simulate the water level oscillation in a better way.

8 Conclusion

The developed hydrodynamic model solves the time dependent depth averaged equations of continuity and motions considering on overlapping finite volumes formed in a three dimensional triangular surface mesh which preserves the geometrical complexities of variable bed topography. As a sample application of the developed model to solve flow patterns in a geometrically complex domain, the tidal currents in GHESHM canal are modelled. The comparison results on water level surface and velocities, demonstrate the ability of the numerical model to tidal current simulation in the marine canals.

References

[1] Vreugdenhil C.B, (1994), Numerical Methods for Shallow water Flow, Kluwer Academic Publisher,

[2] Jameson A, Baker T.J.and Wetherill N.P., (1986), Calculation of Inviscid Transonic Flow over a complete Aircraft , AIAA Paper, 86-0103

[3] Weatherill N.P., A Review of Mesh Generation , Special Lecture , (1996), Advances in Finite Element Technology , Civil-Compress . Edinbrough , PP.1-10

[4] Jia Y. and Wang S.S.Y., (1999), Numerical Model for Channel Flow and Morphological hange Studies, Journal of Hydraulic Engineering, ASCE., 125(9)., 924-933

[5] Wu W., (2004), Depth-Averaged Two-Dimensional Numerical Modelling of Unsteady Flow and Non-Uniform Sediment Transport in Open Channels, Journal of Hydraulic Engineering, ASCE., 130(10), 1013-1024

[6] Jameson A, Baker TJ and Weatherill, (1986), Calculation of Inviscid Transonic Flow over a Complete Aircraft, AIAA 24th Aerospace Science Meeting, (Reno-Nevada), paper-86-0103

[7] Sabbagh-Yazdi S.R. and Mohammadzadeh-Qomi M., (2004), Finite Volume Solution of Two-Dimensional Convection Dominated Sub-Critical Flow Using Unstructured Triangular Meshes, International Journal of Civil Engineering, 2(3), 78-91

[8] Dietrich J.C., Kolar R.L., and Luettich R.A. Assessment of ADCIRC's Wetting and Drying Algorithm School of Civil Engineering and Environmental Science, University of Oklahoma, Norman, 73019,

[9] Thompson J.F. , Soni B.K. and Weatherill N.P., (1999), Hand book of grid generation, CRC Press, New York.,